Aerodynamics of Vehicles in Tubes

D. W. Kurtz

Environmental Sciences Division

Currently many transportation systems are being studied which require vehicles to operate in tunnels under conditions of high blockage. As a consequence, a great deal of interest is being generated in the aerodynamic characteristics of such transportation systems. Model testing should be performed to better understand the aerodynamic aspects of such systems. This article presents a brief description of a facility that was constructed to study these systems and some of the initial results which have been obtained to date.

Introduction

Currently many transportation systems are being studied which require vehicles to operate in tunnels (tubes) under conditions of high blockage (vehicle frontal area occupying a significant portion of the tunnel cross-sectional area). When this condition exists, the vehicle drag can be one to two orders of magnitude greater than the value in free air. In addition, the air velocities in the tunnel ahead of and behind the vehicle can be an appreciable ratio of the vehicle velocity. A proper understanding of these aerodynamic characteristics is necessary in order to efficiently design systems utilizing vehicles in tunnels, which in turn will make it possible to choose an optimum transportation system. In conjunction with theoretical studies by the Aeronautics Department at the California Institute of Technology, an experimental program was initiated at JPL on the aerodynamics of vehicles traveling in tubes.

Facility

Currently operational is a 21.5-m (70-ft)[1] long, 5.3-cm (2-in.) diameter, vertically oriented aluminum tube that utilizes gravity to propel the test

[1] Values in customary units are included in parentheses after values in SI (International System) units if the customary units were used in the measurements or calculations.

Figure 1. Installation of VICS-70 facility

models. This facility, referred to as VICS-70 (vehicles in constrained spaces), and two typical models are shown in Figure 1.

The 21.5-m (70-ft) tube replaces an original 9.8-m (32-ft) long, 5.3-cm (2-in.) diameter, vertically oriented Plexiglas tube that was used to develop the necessary testing techniques for the more advanced, high-precision facility now in operation. The 21.5-m (70-ft) tube is instrumented along its length with magnetic-coil pickups which are triggered when magnets in the model pass each station and indicate model position along the tube as a function of time. This configuration yields data from which model velocity, acceleration, and drag can be determined. Wall pressure ports are provided for recording the pressure history along the

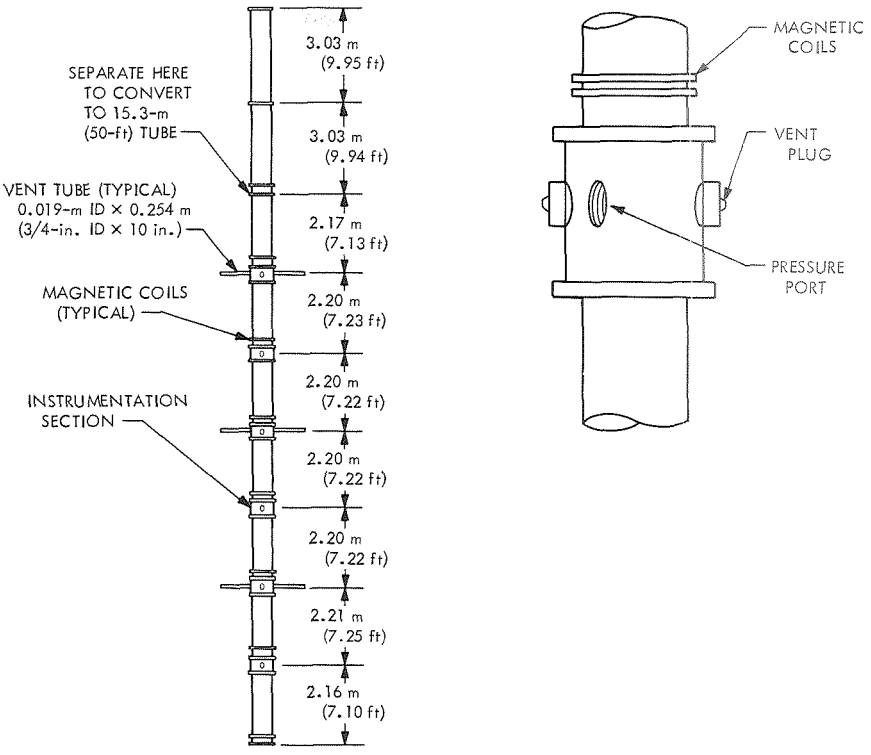

Figure 2. Schematic of VICS-70 facility

tube. The tube is aligned to within ±0.4 mm (±0.015 in.) over the 21.5-m (70-ft) length. A schematic of the tube is shown in Figure 2.

The present facility is a very versatile one; several geometric characteristics of the 21.5-m (70-ft) tube are readily alterable. The tube length can be decreased to 15.3 m (50 ft) or effectively increased up to an infinite length by placing restrictions or orifices at the exit. Tube lengths shorter than 15.3 m (50 ft) require that sections of the tube be removed, and hence necessitate tube realignment upon their replacement. Venting ports are provided along the entire length of the tube. The interior walls of the tube are aerodynamically smooth but controlled roughness of the walls can be introduced. It can be pressurized up to 1.03×10^6 N/m^2 (150 psi) and can use different fluids to obtain data through a wide range of Reynolds numbers. A model launcher is used, when testing at atmospheric pressure, to vary the initial velocity of the test models up to 15 m/s (50 ft/s). Also, the flow velocity in the tube at the time of model launch can be controlled in order that equilibrium test conditions can be obtained as soon as possible. To date, only circular cross-section models equipped with fine Teflon centering skids have been tested. The effects of model eccentricity in the tube and several models traveling simultaneously in the tube can be investigated.

Experimental Results

Drag

The effects of tube length and model blockage on the drag coefficient of the models tested are appreciable. Test section length effects are presented for three blockage ratios in Figure 3. (See Table 1 for definitions of symbols used.) It should be pointed out, however, that some Reynolds number effects are superimposed. Here, the drag coefficient is based on the model velocity relative to the test section wall. The drag coefficient in free air is about 0.5. Model length effects are important but to a lesser degree. This effect is not nearly as strong for higher blockage models and diminishes as models become very long. Nose and tail shapes exhibit distinct, although minor, effects on the drag coefficient. Streamlined nose

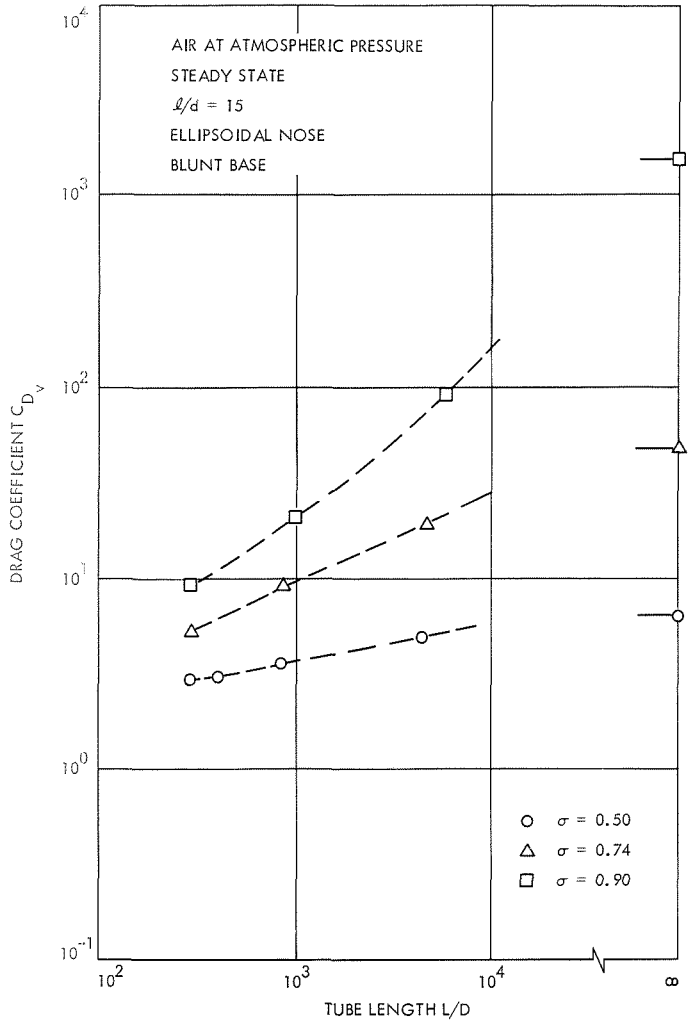

Figure 3. Variation of drag coefficient with tube length for three blockage ratios

Table 1. Nomenclature

a	model cross-sectional area, m^2
d	model diameter, m
C_{D_v}	drag coefficient based on the model velocity relative to the tube, $C_{D_v} = \text{drag} \Big/ \frac{1}{2}\rho v^2 a$
L/D	ratio of tube length to tube diameter
ℓ/d	ratio of model length to model diameter
Re_{d_v}	Reynolds number based on the model velocity and model diameter, $Re_{d_v} = \rho v d/\mu$
v	model velocity relative to the tube, m/s
μ	viscosity of test gas, $N\text{-}s/m^2$
ρ	density of test gas, kg/m^3
σ	ratio of model cross-sectional area to tube cross-sectional area (sometimes expressed in percent)

and tail sections resulted in about a 10% drag reduction on models with a blockage ratio of 0.5 but little or no reduction on models with a 0.9 blockage ratio.

The average velocity of the flow ahead of (and behind) the model increases with the blockage ratio and decreases with the tunnel length. The drag which the model experiences is a direct result of this since it must move (or attempt to move) part of this column of air surrounding it.

The drag coefficient is quite sensitive to test Reynolds number in the lower Reynolds number region (Figure 4). However, the drag coefficient becomes much less sensitive at Reynolds numbers that are near full-scale conditions.

Pressure Signature of Model Traveling in Tube

Experimental investigations performed to date on tube–vehicle systems indicate that the near-field flow is quite complex even for the ideal case; i.e., steady-state, incompressible-flow conditions. The characteristics of the near-field flow (flow extending from just ahead of the model to just aft of the model) can, in part, be inferred from the pressure distribution along the tube as the model passes. The experimental data show the near-field flow to consist of many clearly distinct regions. As few as four and as many as seven of these regions have been observed in each model pressure signature. A sample of a pressure signature at the test section wall in the vicinity of a 50% blockage model traveling at its equilibrium

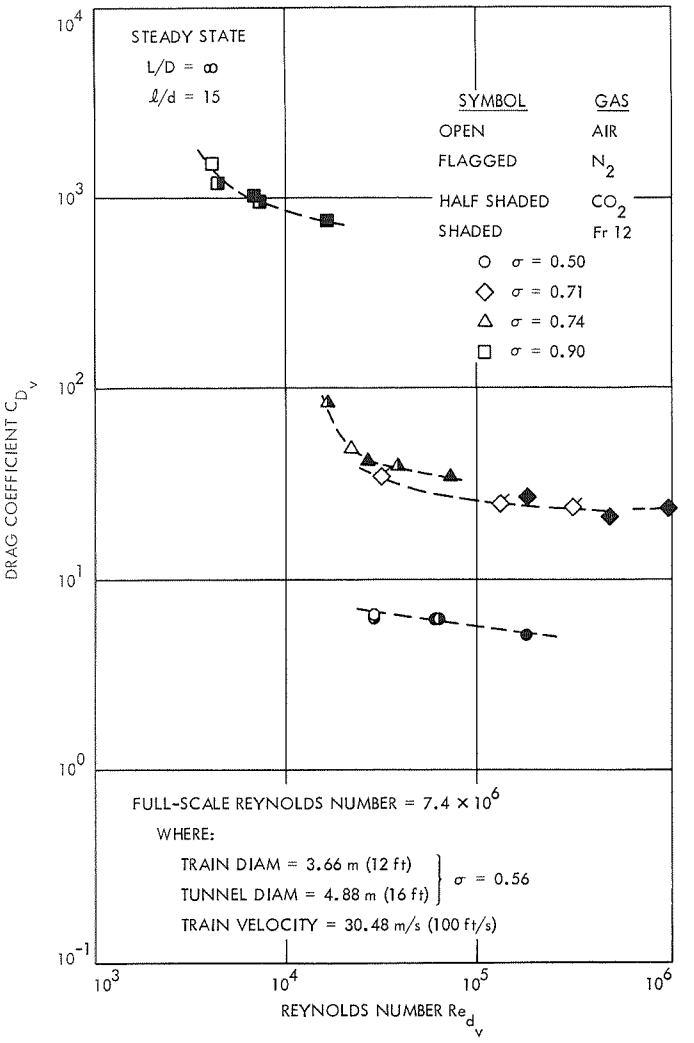

Figure 4. Variation of drag coefficient with Reynolds number for three blockage ratios

velocity is presented in Figure 5. Although recorded on a time scale, the figure has been converted to a pressure distribution along the test section in order to indicate the relative position of the model. Regions 1 and 6 indicate the pressure gradient in the test section due to the flow ahead and behind the model, respectively. Region 1 is essentially pure pipe flow, whereas Region 6 has certain wake effects embedded in the flow. Region 2 is representative of the Bernoulli pressure drop as the flow velocity increases over the nose of the model. Region 3 is the pressure gradient along the model length. Sometimes this occurs as a definite two-slope region, indicating the possible existence of laminar and turbulent flow regimes. Region 4 is usually present but to varying degrees. It is due to an over-expansion of the flow at the model base and acts

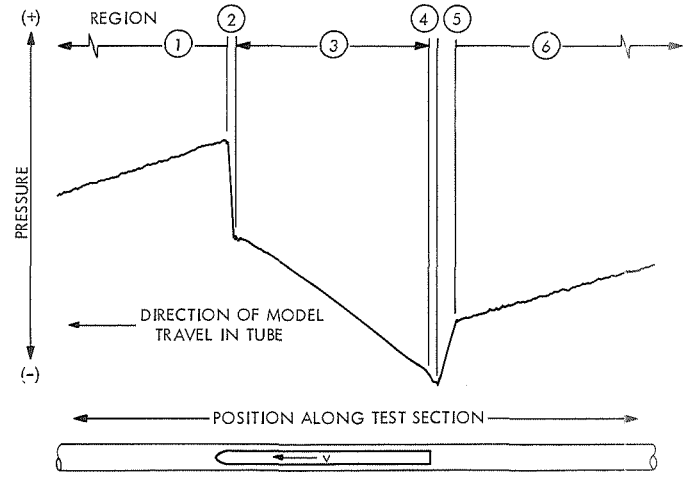

Figure 5. Sample of a pressure distribution along the tube in the vicinity of a 50% blockage model

to increase the drag. Region 5 is the pressure recovery as the flow velocity decreases to downstream conditions.

A great deal of useful information can be obtained from the model's pressure signature, such as:

(1) Velocity of flow in tube.

(2) Bernoulli pressure drop over model nose.

(3) Effective flow skin friction coefficient along model and tube immediately surrounding model.

(4) Pressure drop of flow expanding over base of model.

(5) Pressure recovery in flow downstream of model base.

(6) Effect of model wake on pipe flow friction factor.

The characteristics of the model pressure signature are greatly affected by the model blockage ratio. The tube length affects the signature to a smaller degree. For the 50% blockage model, there is both a relatively large initial pressure drop and a large pressure recovery. For the 90% case, the pressure gradient due to friction along the model and tube is the dominant characteristic of the pressure signature.

The pressure drop over the nose of the vehicle is about as would be expected from simple Bernoulli considerations for the 50% blockage model, but becomes increasingly larger than expected as the model blockage increases. The pressure recovery downstream of the model decreases as the model blockage increases in a manner not far off from the decrease expected from strict theoretical considerations.

Effective friction factors on the model and along the tube wall immediately surrounding the model can be inferred from the pressure signature of the model. Based upon the limited data analyzed to date, it appears that the concentric pipe flow approach is better than the flat plate approach. Not enough data have been analyzed to determine if there are significant effects on the friction factor due to the variables investigated (tube length, model length, model blockage, and gas).